

Lab experience 2: Introduction to AutoCAD Inventor

Inventor and SolidWorks are solid modeling software environments, with which we can draw, manipulate and analyze a variety of useful objects, or *models*.

The models we draw with SW and Inventor are used for a variety of purposes:

- To make illustrations of our designs and devices
- To create engineering drawings that manufacturers can use to fabricate our products
- For rapid prototyping with a 3-D printer
- As inputs to physical modeling and structural animation
- For export to COMSOL or other engineering analysis software.

In BIOEN 327, we are primarily interested in illustrations and 3-D printing.

PART I: FAMILIARIZATION

Where to find the software

Inventor is installed in the Advanced Computing Lab (Foege N140). You may also obtain the software for your own use via free download:

<http://www.washington.edu/itconnect/wares/uware/autodesk-for-education/>

How to start the software

I bet you can figure this out for yourselves, but just for completeness...

Start button >> All programs >> Autodesk >> Autodesk Inventor

Suggested tutorial videos

The Inventor software includes a set of tutorial videos that can be accessed via the drop-down Help menu > Onboarding Videos. Here are my comments and suggestions about the videos:

Overview

Overview and *View Navigation* are useful. We will use only parts and assemblies, not drawings.

The *Work Features* will be useful once you know more about the software so come back to it later.

Skip the *AutoCAD Import*.

Sketches

The *Get Started with Sketch* video has lots of good information but goes too quickly!

In Tutorial 1: *Sketch Basics*, do steps 1-5. Steps 6 & 7 are optional.

In Tutorial 2: *Navigate the Sketch Environment*, has some good tips in step 1, but the rest seems a little advanced for us today.

Tutorial 3: Use sketch to build a part. If you can do all of these steps, you know how to sketch pretty well. Get through as much as you can.

Parts

Watch the first section of the *Parts* video. We will probably not use sheet metal or multi-body parts BIOEN 327.

In Tutorial 1: *Part Basics*, do Step 2; the others are optional. Tutorials 2 and 3 are also optional.

Assemblies

Watch the *Assemblies* video. It shows many of Inventor's coolest capabilities. Unfortunately, the tutorial steps themselves assume that you have gone through the rest of the tutorials in detail and I do not recommend them at this time.

So... now you have a choice.

If you want to learn more about the Inventor commands by reading, go to Help (pull-down menu) > Learn more > Learning path. That resource provides a text-based overview of the various functions that you can use in sketches, parts and assemblies.

If you want to learn as you go, try the Lab 2 assignment. It assumes that you know the basics of sketches, parts and assemblies, and gives some hints to save you time searching in the Help.

Lab 2 Assignment : BIOMEDICAL MODEL

In this exercise you will create an Inventor (AI) model of a hip implant, or of another implantable structure of similar complexity. A hip implant consists of three main parts: the stem or spike that fits into the femur, the ball that is permanently affixed to the stem, and the polyethylene cup that fits into the pelvis.

Drawing the metal portion of the implant in AI follows a procedure often used to fabricate the spike out of metal. A flat plate of stainless steel or titanium is cut with a water jet to make a sharp-edged basic shape. The edges are then rounded by a variety of machining techniques. The ball is created separately out of the same material, and affixed to a post on the upper end of the spike. The metal surface is polished mechanically and then polished electrochemically. Similarly, in AI we will make a flat “boss” in the basic shape of the spike, then fillet the edges to produce the desired smooth model. The ball will be created as a separate part and the two will be joined in one Assembly. We will not electropolish the drawing, but it is possible to add surface effects including colors and textures. A [beautiful, although not surgically ideal, rendering](#) can be found on the grabcad.com site.

Normally an engineering drawing or solid model is made as part of a larger engineering process that includes defining your need, establishing performance characteristics (specifications), considering constraints, proposing a design (which includes the drawing), building a prototype, testing it, and refining your design. However, for today we are simply introducing AI so we will use dimensions that have been published or measured from an existing hip implant.

Step 1: Plan your drawing

It is tempting just to start drawing and figure out the dimensions as you go, but it is a good practice to plan your work by sketching the device by hand in your lab notebook. You get to consider options and make notes in a quick and informal way before formalizing the layouts in AI. Having that sketch on a dated notebook page also shows that the design is your work and that it was created at the time you claim, which is important if you would like to patent your device. In this exercise, follow these steps:

- 1a. Determine the dimensions you need. These dimensions may be taken from an actual implant (the lab has a used specimen) or from sources such as manufacturers' literature.¹
- 1b. Plan the division of parts in the assembly.
- 1c. Sketch each part and label with the appropriate dimensions. Remember that some dimensions depend on others, so the order in which you apply the dimensions matters.

¹ <http://www.zimmer.com/content/dam/zimmer-web/documents/en-US/pdf/surgical-techniques/hip/zimmer-ml-taper-hip-prosthesis-surgical-technique.pdf>

The Zimmer web site lists the stem/spike as Femoral Components and the ball & cup as Acetabular Components. Zimmer's brochures provide a lot of information about the acetabular shell (metal cup) and liner (polyethylene cup that goes inside the shell) but the dimensions are difficult to extract from their ordering information.¹ You may assume that the shell and liner are each 5 mm thick and that the ball that is fixed to the stem is 30 mm in diameter.

Note that Zimmer has a variety of implant types, and you may choose a knee or shoulder implant if you prefer.

Additional references:

<http://www.omicsonline.org/scientific-reports/srep309.php> (Stress analysis of various spike and neck and cup angles)

<http://www.ncbi.nlm.nih.gov/pubmed/15348867> (study of cup thickness and head diameter and fusion effectiveness, including trials with irradiated cups. Includes specimens recovered from cadavers. Thickness of cup is about 10 mm.

¹ <http://www.zimmer.com/content/dam/zimmer-web/documents/en-US/pdf/surgical-techniques/hip/trabecular-metal-modular-acetabular-system-surgical-technique.pdf>, page 9.

Step 2: Drawing the hip implant stem as a part

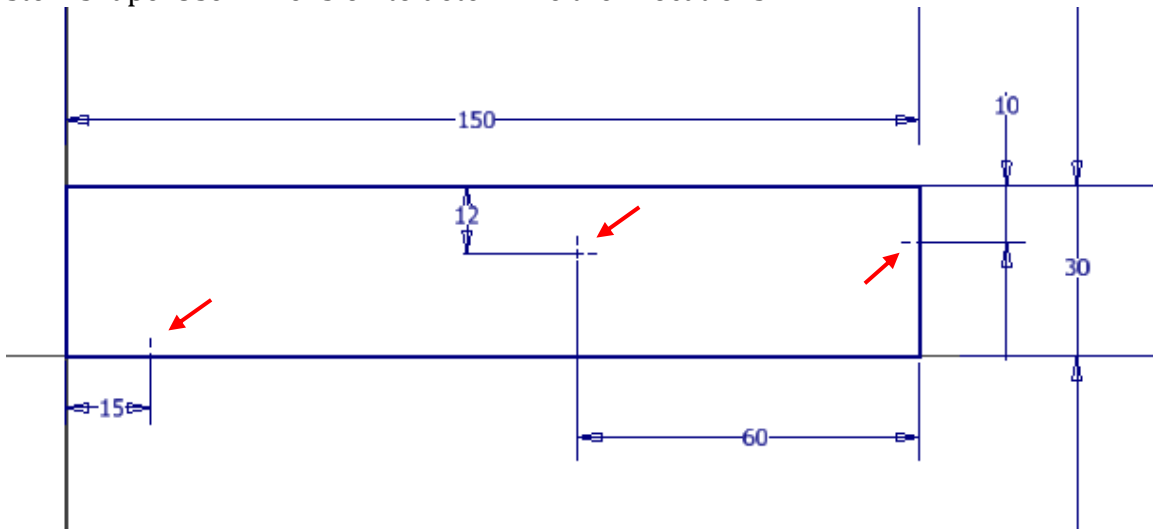
The following procedure is one of many that you can use to create an implantable stem of approximately the same shape as in the Zimmer documentation. It is not meant to be a tutorial in how to use Inventor, but rather a guide for one specific application. Your dimensions do not have to match the dimensions shown below.

Tip 1: make sure you are using the units you want. You may change the units via Tools >> Document Settings >> Units.

Tip 2: I have hidden the constraints (right click > Hide All Constraints) so by default your sketch will have additional icons. You may choose to hide or show as you like.

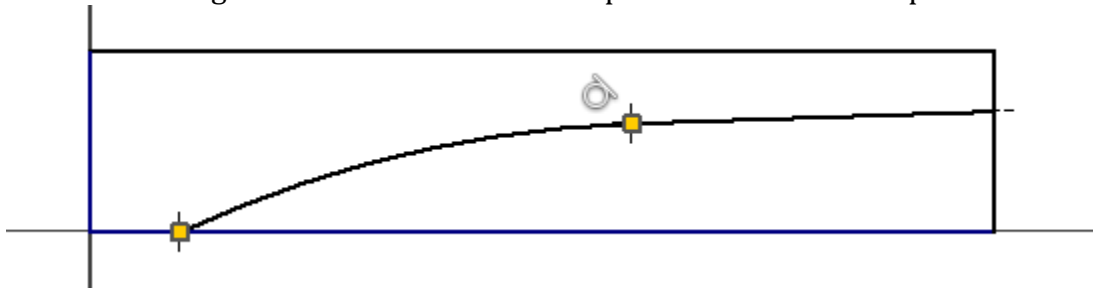
Create a two point rectangle with the lower left corner at the origin; add dimensions either before you left-click, or by using the Dimension tool (D).

Add three points that will define the curve that is cut out of the rectangle to make its stem shape. Use Dimension to determine their locations.



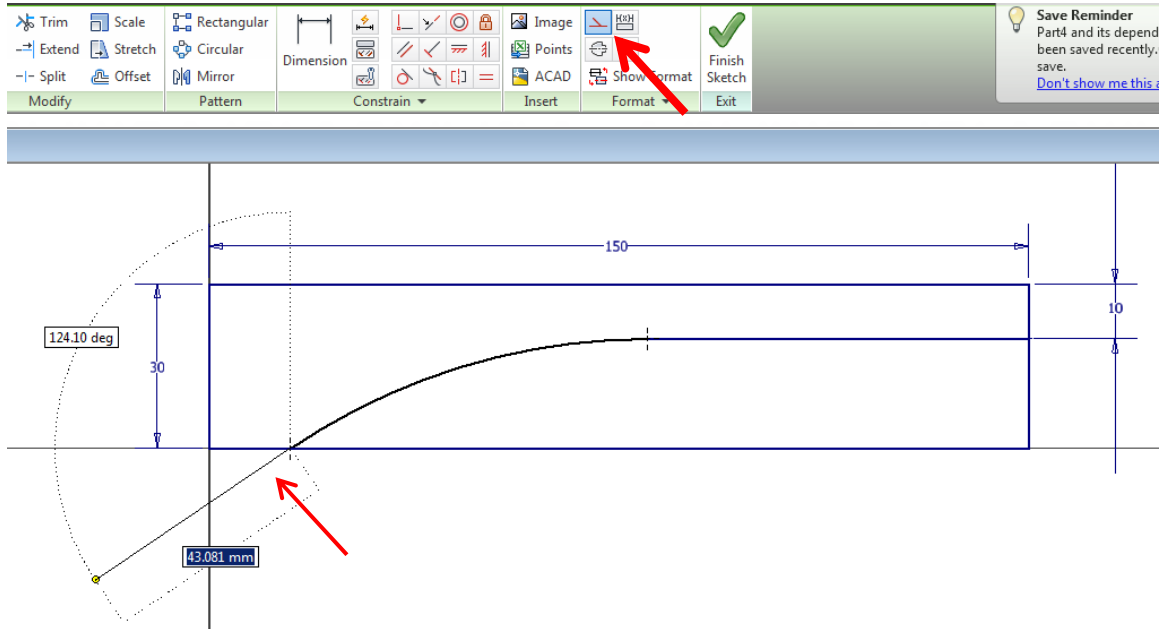
Draw a line between the right two points.

Select Arc>Tangent and connect the middle point to the lower left point.

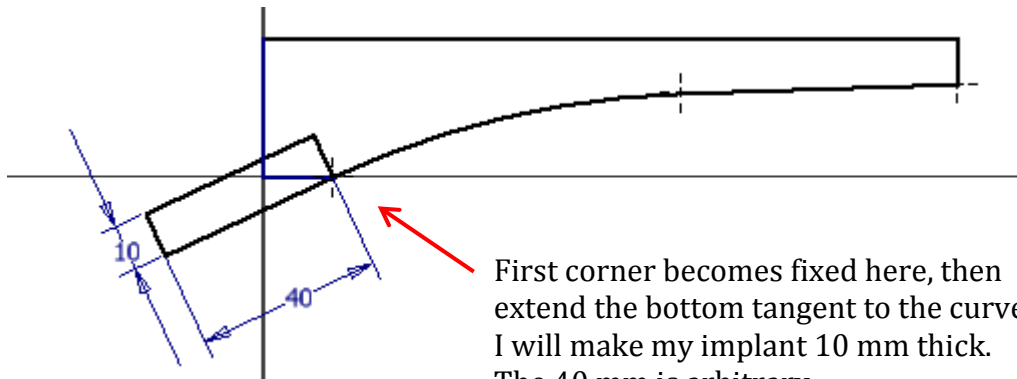


Use Trim to delete the lower right corner of the rectangle. Just click Trim and then click on each segment you want to delete.

The next step is to add a rectangle that will become the stem onto which the ball fits. As setup, I suggest making a construction line that is tangent to the tangent arc you just created. Construction lines show up as dashed lines, and give you reference locations, but do not form part of the outline in your finished sketch. To make a construction line, click the for construction icon, then start a line at the end of the tangent arc, then adjust the angle until it snaps to being tangent.



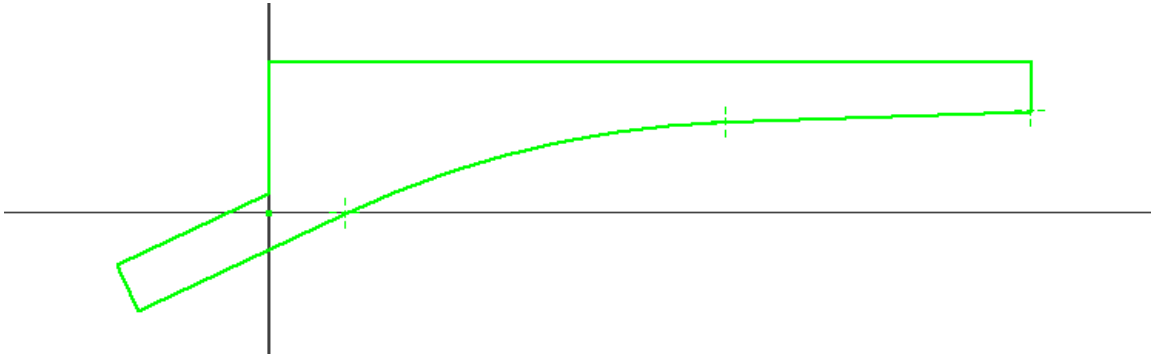
Select Rectangle > 3 Point and draw a rectangle that protrudes off the lower left of the stem. After I clicked on the first (right-hand) corner, I extended the bottom edge of the rectangle along the construction line and set its length, then clicked and extended its width. The length of the rectangle doesn't matter much, but its width should be the same as the thickness of the whole piece when you extrude after sketching (thickness = the distance out of the plane of the sketch).



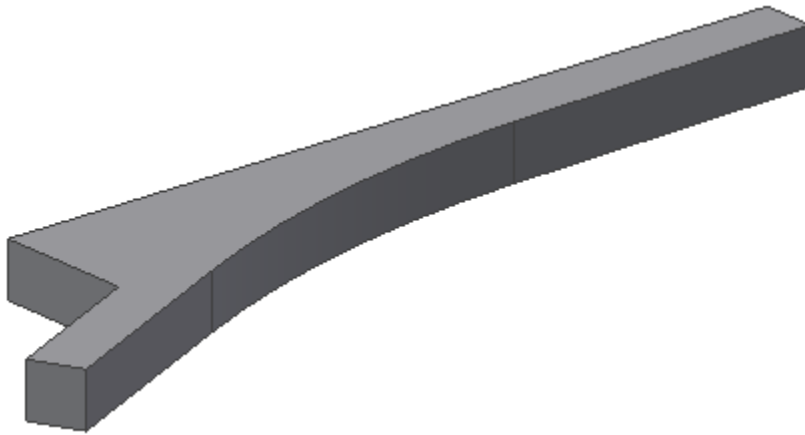
First corner becomes fixed here, then extend the bottom tangent to the curve. I will make my implant 10 mm thick. The 40 mm is arbitrary.

Use Trim to delete the inner segments. An alternative to Trim is to right click on a segment and Delete. However, Delete might take out an entire side of a rectangle so sometimes Trim is the best option.

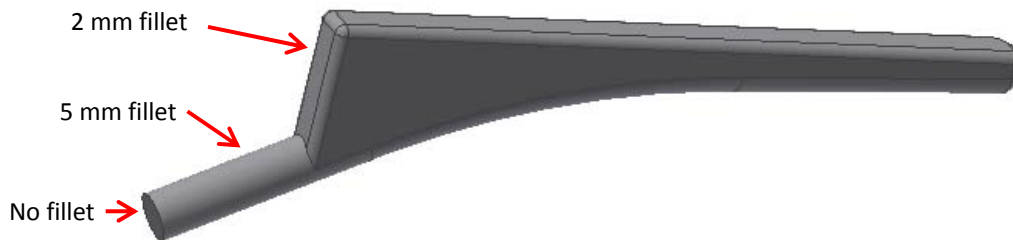
The sketch is now done. Note that I also selected all of the lines, right-click > Properties and changed the line color to make it easier to see over the black axes.



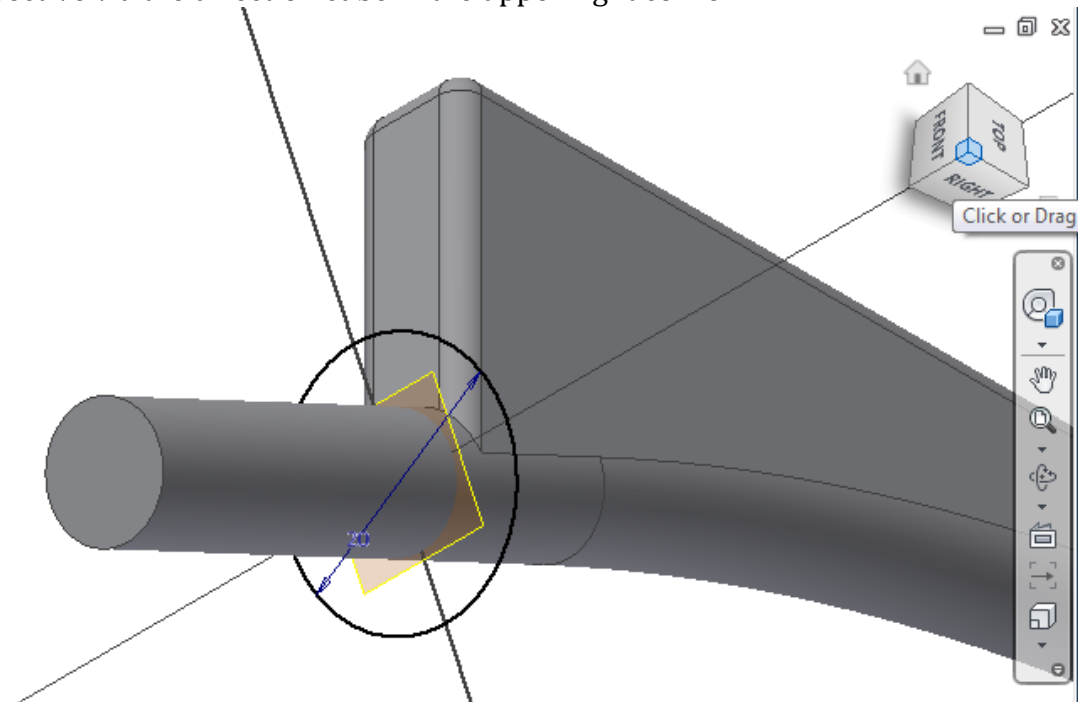
Click Finish Sketch. Click Extrude and enter (or keep) the distance you want. Click OK. I then went to the View tab > Visual Style > Shaded with Edges to give the figure shown here. Now save the model!



Add small-radius fillets to all edges except on the neck (the exact radius doesn't matter). Add fillets to the four side edges of the neck (not the perimeter of the resulting circle at the end of the neck). These fillets should have a radius half of the thickness of the extruded piece. This large fillet propagates along the inner curve -- don't worry about it.



If you like, you may add a disk that will rest atop the femur and keep the stem from sliding farther into the femur. To do this, you will need to draw a circle that is parallel to, but offset from, the end of the neck (the cylindrical portion). Under the 3D Model tab, click Plane, then select the end of the neck. Enter the offset distance; on mine this was a negative number. This gives you a sketch surface that is offset from the end of the neck. Start a 2D Sketch on this offset plane. This will, by default, turn that surface toward you. However, if you like you may choose another perspective via the direction cube in the upper right corner.

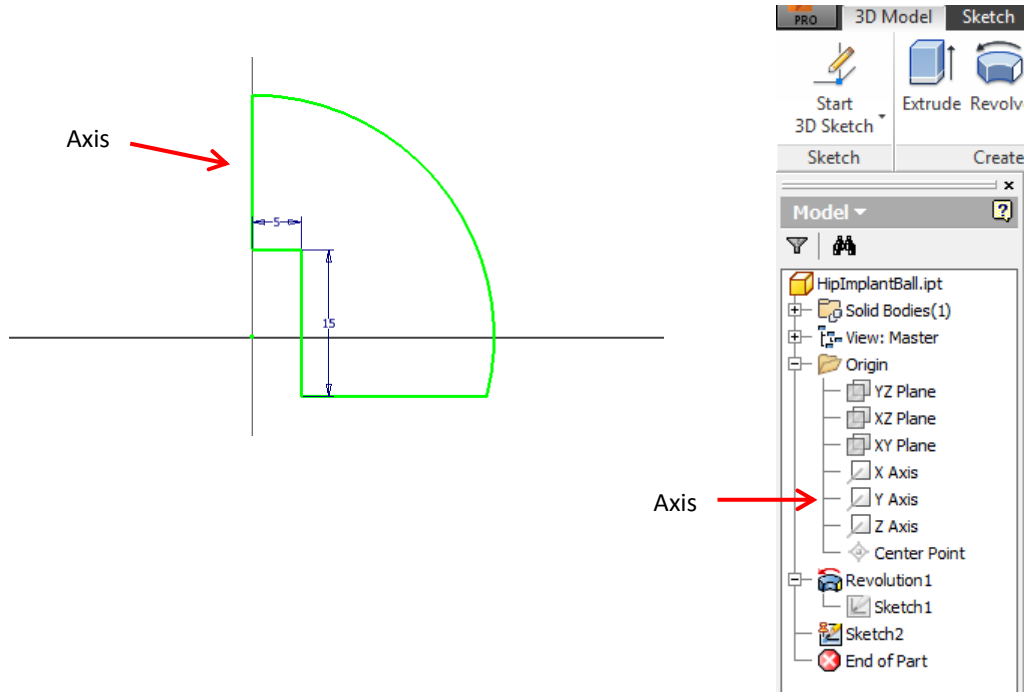


Draw a circle with its center at 0,0 in this sketch plane. Set the diameter you desire, then Finish Sketch. Select and extrude the circle. Add fillets to the disk, and you are done with the stem!



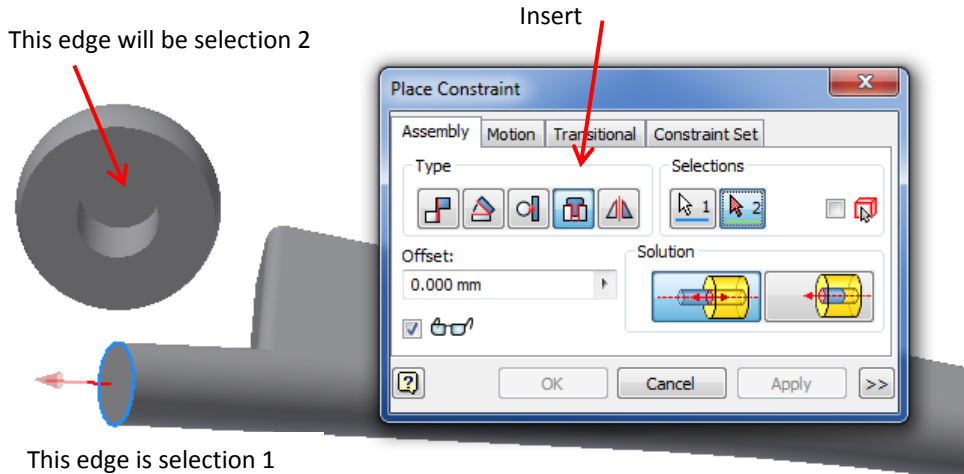
Step 3: Draw the ball as a part

There are a few ways to make a ball that is flat on one side and has a hole drilled in it. One is to make a sketch with half of the cross-section, Finish Sketch, then Revolve it 360°. While using the Revolve tool, the green outline below is the Profile, and the vertical line is the Axis. If the sketch does not already include a line that is your desired axis, then you may either draw a line for this purpose, or use the X, Y or Z axis as appropriate. These axes are accessible in the Model tree, under Origin.



Step 4: Combine the parts in an assembly.

After creating a new empty assembly, Place the two parts that you have previously saved. Their orientations depend on the plane(s) you chose for making your sketches back when you created the parts. You want to affix the ball on the stem, so we Constrain its location there and choose the Insert option.

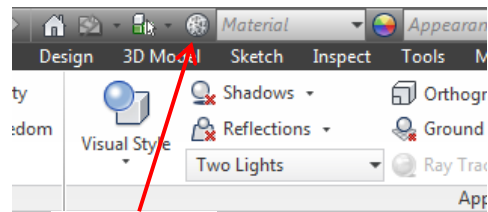


Step 5: Draw the polyethylene cup as a part.

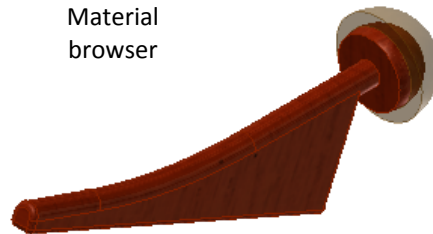
The cup is a hemispherical shell whose inner surface matches that of the ball and whose thickness is 5-10 mm.¹ Construction is similar to making the ball, but of course you sketch two concentric arcs before rotating the sketch. The concentric arcs may be generated individually, or by using the Offset function in the sketch.

Step 6: Place the cup part in the existing assembly, and this time use a Joint to combine the two. The type will be Ball, and the two surfaces will be the outside of the ball and the inside of the cup.

Step 7: Use the material browser to apply a surface appearance to the parts in your assembly. Select either the cup or the metal assembly, scroll to the material you like and right-click > Assign to selection.



Material browser



Step 8: **Work submission.** Collect your parts and assemblies (.ipt and .iam files) and upload them to the Catalyst or Canvas drop box. Show the instructor or TA that you have used your notebook to make draft sketches for your parts.

¹ [J Mater Sci Mater Med.](#) 1998 Jul;9(7):393-401.